



Quick Start Guide  
**Simplify3D V5**  
Food printing manual

[www.FELIXfood.nl](http://www.FELIXfood.nl)  
[support@felixprinters.com](mailto:support@felixprinters.com)

Zeemanlaan 15  
3401MV IJsselstein  
The Netherlands

# 1 Introduction

The goal of this guide is to explain how to use Simplify3D V5 (slicer) together with the FELIX FOOD printer and ensure you get successful prints as quickly as possible.

The following topics are covered:

- Installation of software
- General 3D printing process
- Preparing the printer and software for 2 example cases
  - Single head (for Single, Twin and Switch)
  - Dual head (for Switch)
- Configure new recipes
- Tips and tricks

For more in depth information about Simplify3D or 3D printing in general, you can use the following resources:

- [www.felixfood.nl](http://www.felixfood.nl)
- [www.Simplify3D.com](http://www.Simplify3D.com)

If you are unable to continue with your printjob or have any questions, you can check the support section of our website or you can contact us directly:

- **Website:** [www.felixfood.nl/contactus](http://www.felixfood.nl/contactus)
- **Email:** [support@felixprinters.com](mailto:support@felixprinters.com)
- **Telephone:** +31 (0)30 30 31 387
- **Address:** Zeemanlaan 15, 3401MV IJsselstein, The Netherlands

# 2 Table of Contents

1	Introduction.....	1
3	3D Printing Process .....	3
4	Simplify installation.....	4
5	Slice profile installation.....	5
6	Slice profile setup .....	6
7	Single head printing .....	7
7.1	Import .STL file .....	7
7.2	Aligning the models .....	8
7.3	Slicing a model .....	9
7.4	Save printfile (single).....	10
8	Dual head printing (Switch only).....	11
8.1	Import STL File .....	11
8.2	Multi extrusion wizard .....	12
8.3	Slicing the models .....	13
8.4	Save printfile (dual) .....	14
9	Printing with support .....	15
10	Adding new recipes .....	16
10.1	Add a new material .....	16
10.2	Configure new material settings .....	17
11	3D printing tips & tricks .....	20
12	3D printing design tips .....	21

# 3 3D Printing Process

In general the 3D printing process from design to printed object can be described as follows:

## 1. Import CAD file



Simplify3D software generally accepts STL files.

## 2. Prepare printer and software parameters



Prepare the printer and software parameters for the application.

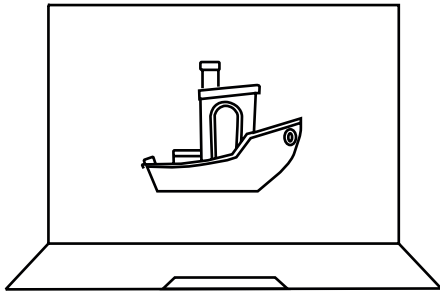
## 3. Slice and inspect



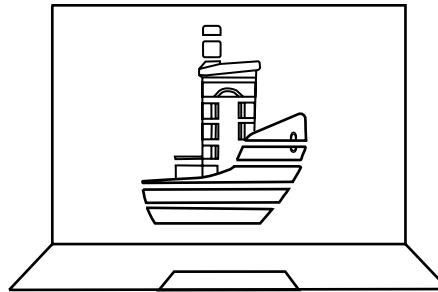
The object is digitally cut in thin layers and paths are calculated per layer for printing.

## 4. Print

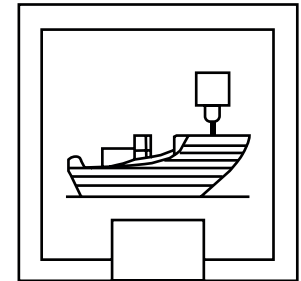
The printer will extrude material on the calculated path, layer by layer, until you have a finished product.



**STL FILE**



**G-CODE**



**PRINT**

# 4 Simplify Installation

- 1 >> Download and Install simplify3D, follow the instructions on the supplied voucher card to activate your product.
- 2 >> Run simplify3D
- 3 >> Press cancel when the configuration assistant pops-up after the startup of the program. (Image 1)
- 4 >> Use the printer settings (slice profiles) provided by FELIXprinters. They are most up to date and give the best print results.
- 5 >> Get slice profiles from the SD card or the download section:
  - SD card provided with the printer.
  - Go to folder Software\Simplify3D
  - Locate the \*.fff file corresponding to your printer.

Download section FELIXprinters website ([www.felixprinters.com/downloads](http://www.felixprinters.com/downloads))

- Go to folder: 3. Software ▶ 1. Simplify3D ▶ 4. Sliceprofiles. (Image 2)
- Choose the correct slice profile and download it to your device with simplify3D installed.

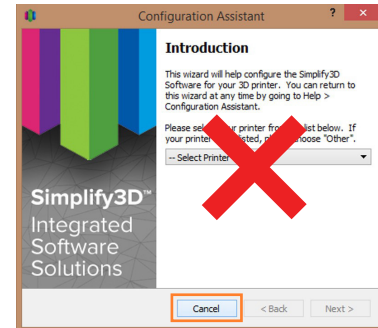


Image 1

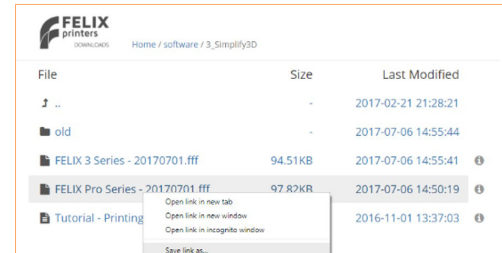


Image 2



KEEP IN MIND TO USE THE RIGHT SLICE PROFILE FOR THE FOOD PRINTER. THIS PROFILE CANNOT BE FOUND WHEN INSTALLING SIMPLIFY AND NEEDS TO BE MANUALLY ADDED AFTER INSTALLATION. THIS CAN MAKE OR BREAK YOUR MACHINE.

# 5 Slice profile Installation

- 1 >> Open Simplify3D
- 2 >> Choose File --> Import FFF profile (Image 4)
- 3 >> Navigate to the .fff file for your desired printer and open it. (Image 5)
- 4 >> If you correctly imported the Simplify3D profile for the FELIX FOODprinter you will see this message pop-up. (Image 6)

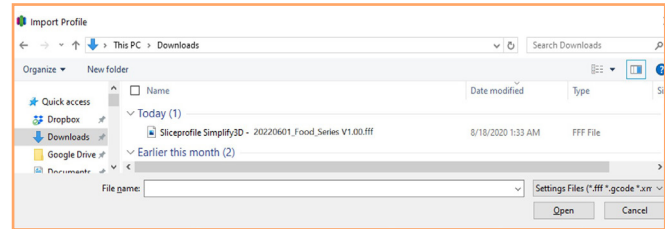


Image 5

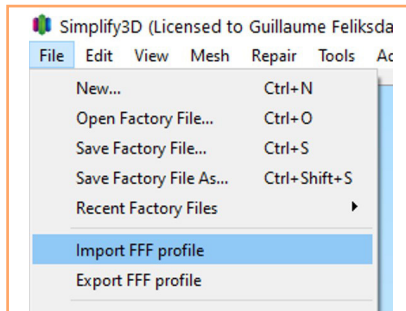


Image 4

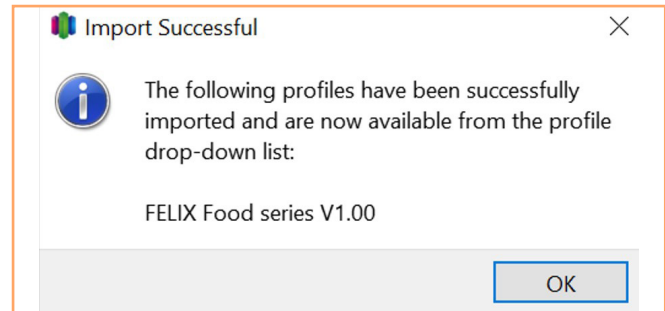


Image 6

# 6 Slice profile Setup

- 1 >> Press the “Add” button to create a new print process. In this window you can configure the printing parameters for your file. (Image 7)
- 2 >> Select the correct profile for the FOOD printer.
- 3 >> Choose the correct printer model. (Image 8 & 9)
- 4 >> Press the “OK” button to finish the profile setup.

PROFILE	APPLICATION
Single	For Single head printer, printing 1 object at a time.
Twin	For Twin head printer, printing 2 objects simultaneously.
Switch Left only	For Switch head printer, when printing with the left syringe only (default for Switch head).
Switch Right Only	For Switch head printer when printing with the right syringe only.
Switch Dual	For Switch head printer when printing with both syringes in one print (Dual head printing).

Image 9

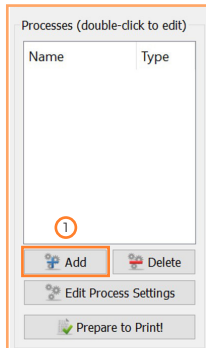


Image 7

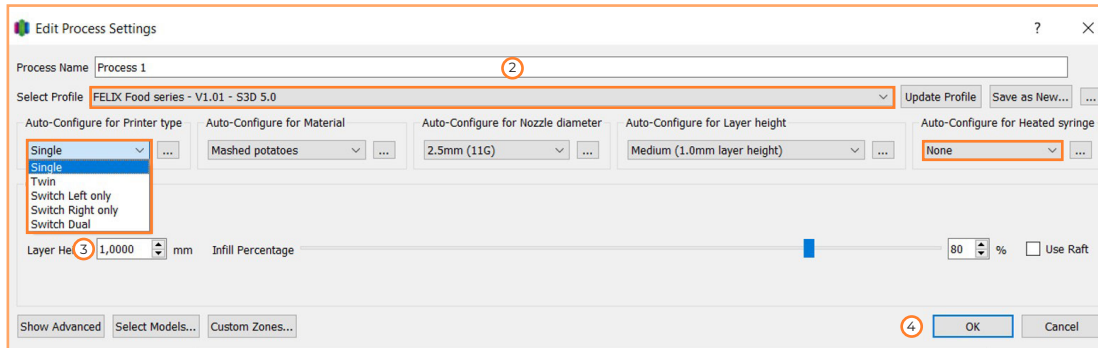


Image 8

# 7 Single Head Printing

## 7.1 Import .STL file

- 1 >> On the left side of simplify3D interface, press the import button to import an STL file. (Image 10)
- 2 >> In this example, we take the “Felix logo.STL”. Locate this file on the supplied SD card or get it of the FELIX FOODprinter website. (Image 11)
- 3 >> Once loaded in, a preview of the STL file will appear in the display. (Image 12)





 02. FELIXFoodV1.5 - Calibration Lines.STL
 02. FELIXFoodV1.5 - Felix Logo.STL
 02. FELIXFoodV1.5 - Mandala.STL
 02. FELIXFoodV1.5 - Shooting Stars.STL

Image 11

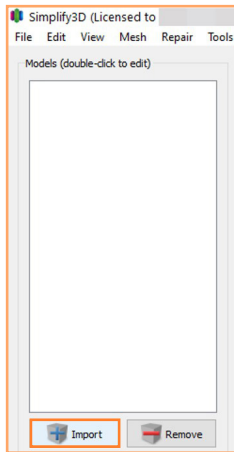


Image 10

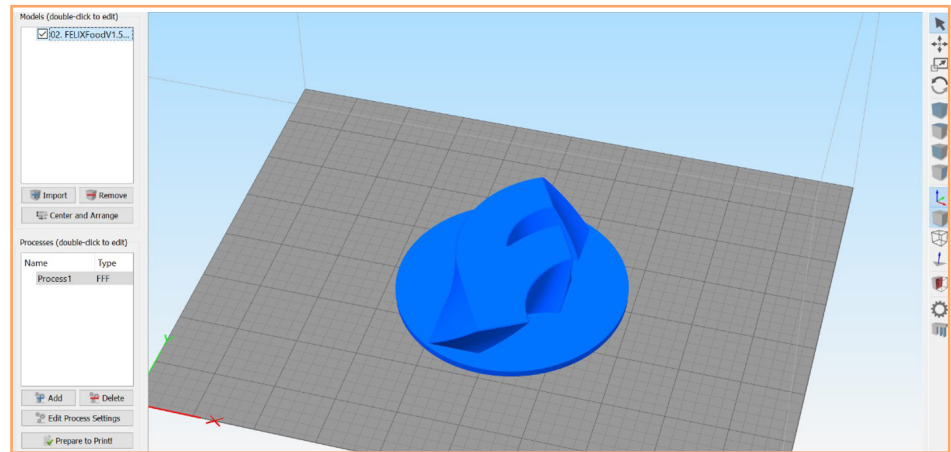


Image 12



## 7.2 Aligning the models

There are several options and the corresponding hotkeys to align your models on the printbed.

- 1 >> Tap the **"Center and Arrange"** to center the model in the middle of the buildplate. (Image 21)
- 2 >> Press **"W"** to open the move menu. You can use the arrows or drag the model to the desired position. (Image 22)
- 3 >> Press **"R"** to open the rotation menu. (Image 23)
- 4 >> Press **"Ctrl + L"** to select a surface that you would like to lay down flat on the buildplate. (Image 24)
- 5 >> Press **"Ctrl + Z"** to undo the previous action.

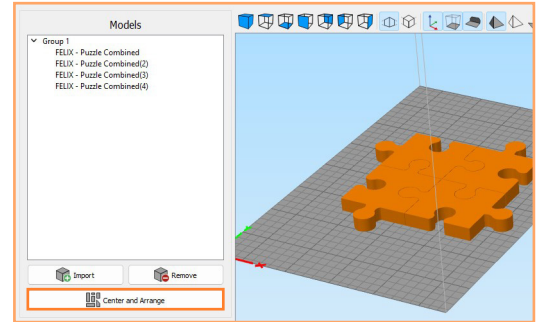


Image 21

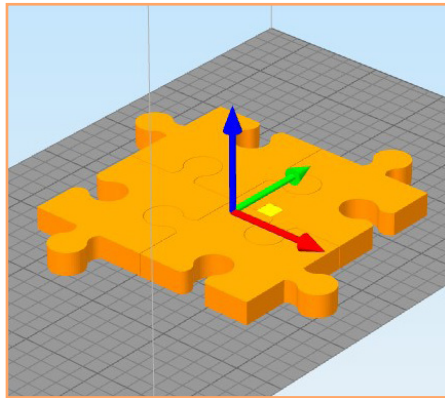


Image 22

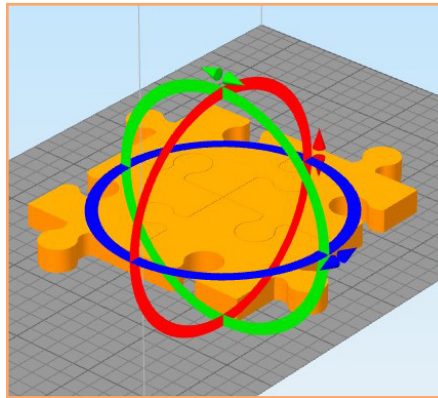


Image 23

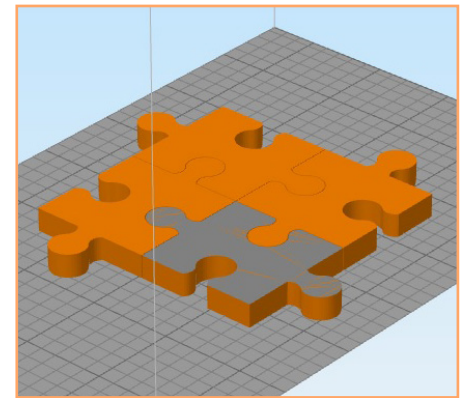
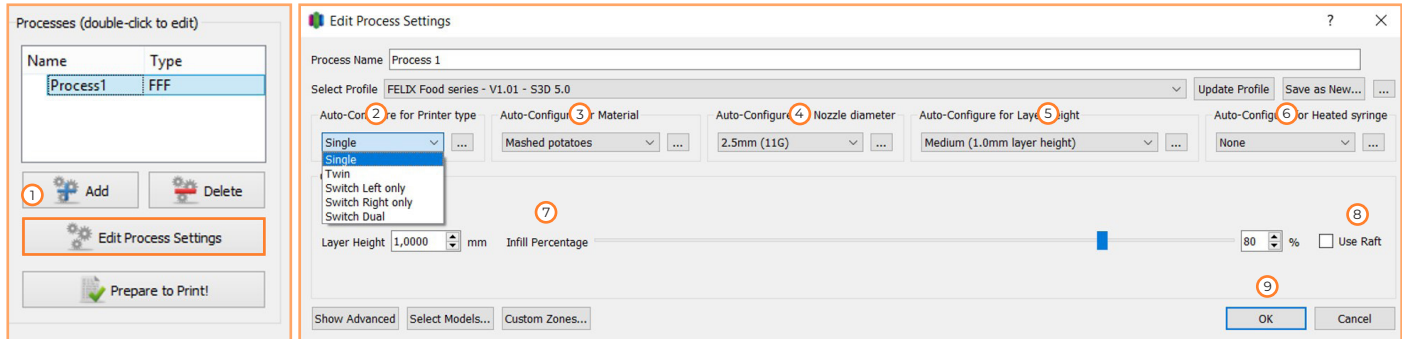


Image 24

## 7.3 Slicing a model

When the .STL file is successfully imported, the object needs to be sliced in layers. Before we start slicing, we have to ensure that the correct process settings have been set up.

- 1 >> Click “**Edit process settings**” in the bottom left corner of the main window.
- 2 >> Select the printer type to match the printer you use. (We will cover dual head printing on page 10).
- 3 >> Select the material type to print with. (We will cover creating your own recipes on page 16).
- 4 >> Select the nozzle diameter to match the nozzle you have installed on your syringe.
- 5 >> Select the layer height. The different options define the layer height for each layer (MAX layer height = 80% of the nozzle diameter).
- 6 >> Select the heated syringe if you have a heated syringe installed on one (or both) syringe holder(s).
- 7 >> Select the amount of infill (0% = hollow, 100% = fully filled).
- 8 >> Raft is an additional thick layer between part and build surface and mostly not used when printing food.
- 9 >> Press OK to continue.



## 7.4 Save print file (single)

- 1 >> Tap the “prepare the print” button. (Image 13)
- 2 >> In the following window you can see (Image 14)
  - >> Printing time estimation.
  - >> Filament/Syringe length used.
  - >> Amount of material usage
  - >> Material costs (if specified)
- 3 >> Different preview line colours.
- 4 >> Inspect the preview of the sliced object and verify that it all looks good.
- 5 >> Save the print file (.gcode) locally or to a microSD card.

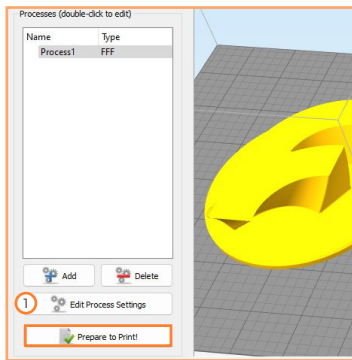


Image 13

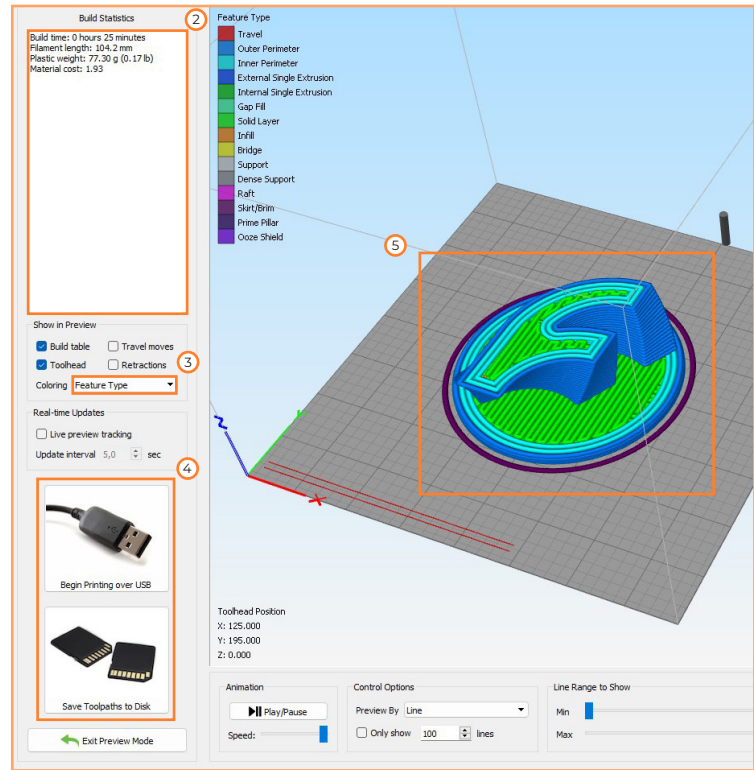


Image 14

# 8 Dual Head Printing (Switch only)

## 8.1 Import STL File

- 1 >> Tap the “import” button to import an STL file (Image 15).
- 2 >> In this example we’re using the “FELIX - puzzle combined”. The files can be found on the provided USB stick (Image 16).
- 3 >> Select the model and tap the “mesh” button in the top menu of simplify. Tap the “seperate connected surfaces” button. This will split the model in separate pieces so we can print the parts individually (Image 17).
- 4 >> After importing the files into the slicer a preview of the model will be shown. We will change the orientation in the next steps (Image 18).

FELIX - Puzzle Combined	28-11-2022 16:26	3D Object	200 kB
FELIX - Puzzle Piece 1	28-11-2022 16:27	3D Object	51 kB
FELIX - Puzzle Piece 2	28-11-2022 16:27	3D Object	50 kB
FELIX - Puzzle Piece 3	28-11-2022 16:27	3D Object	50 kB
FELIX - Puzzle Piece 4	28-11-2022 16:27	3D Object	50 kB

Image 16

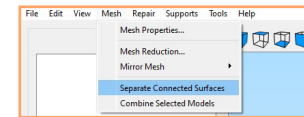


Image 17



Image 15

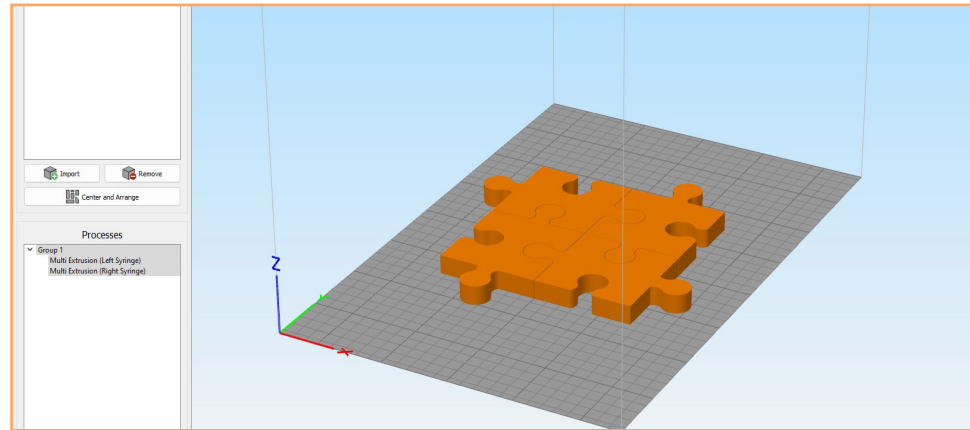


Image 18

## 8.2 Multi extrusion wizard

To setup the dual material prints, we are using the dual extrusion wizard to make the setup proces easier.

- 1 >> Tap the **“Tools”** button in the top bar of simplify3D and tap the **“Multi extrusion wizard”** to open the wizard (Image 19).
- 2 >> Select the food base profile (Image 20).
- 3 >> Select **“Switch Dual”** under auto-configure for printer type.
- 4 >> Select the material/recipe that you would like to use. (new recipes on page 16)
- 5 >> Select the nozzle size that you would like to use.
- 6 >> Select the layer height. **NOTE:** The layer height cannot be larger than 80% of the nozzle diameter. (ratio factor 1,20)
- 7 >> Drag the print files to the left or right extruder of your choice.  
In this case we’re going to print puzzelpiece 2 & 3 with the left syringe and print puzzle piece 1 & 4 with the right syringe.
- 8 >> Check the **“align models origins”** and the **“group models”** checkbox.
- 9 >> Tap the **“Ok”** button.

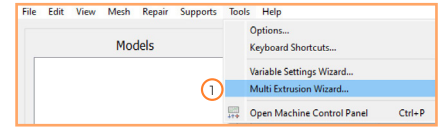


Image 19

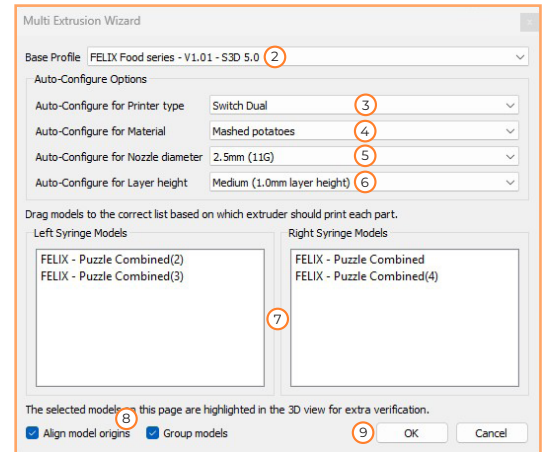
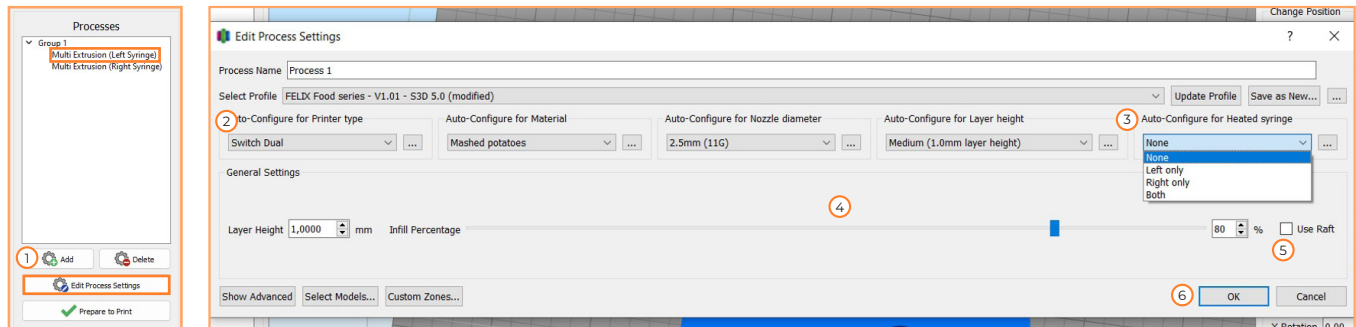


Image 20

## 8.3 Slicing the models

After performing the multi extrusion wizard, the next step is to verify and change the last paramters before exporting the Gcode file to the printer.

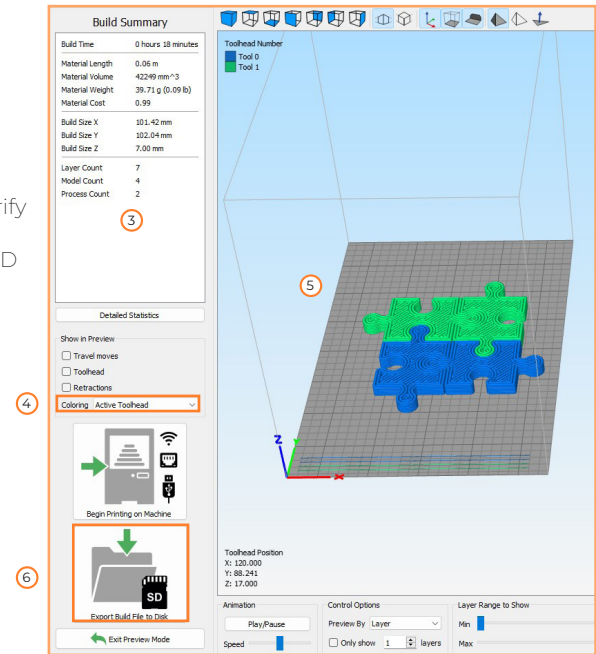
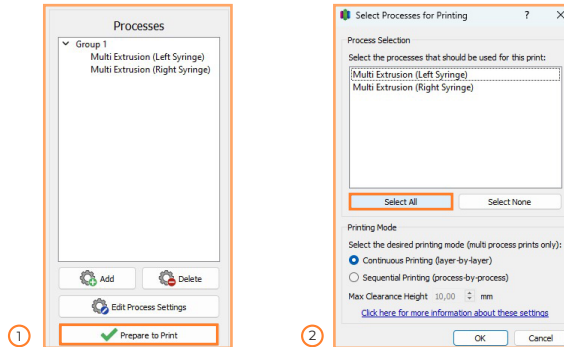
- 1 >> Select the left syringe under processes and press “edit process” at the bottom of the window.
- 2 >> Make sure you don't change the printer type from this point on!
- 3 >> If you're planning on using a heated syringe, tap the “Heated Syringe” to enable it.  
**NOTE:** the heated syringe is an optional upgrade and is not installed as standard on the printer.
- 4 >> Select the amount of infill by sliding the infill percentage bar. (0% = hollow & 100% = solid)
- 5 >> A raft is an additional thick layer that is used in Plastic FDM to ensure good bed adhesion. When printing with food in 9/10 cases you don't need a raft.
- 6 >> Tap the “OK” button to continue.
- 7 >> Repeat the steps for the right extruder.



## 8.4 Save print file (dual)

Once the parameters for both extruders have been setup, we continue by checking the preview of the G-code and adjusting details if necessary.

- 1 >> Press "prepare to print".
- 2 >> Press "Select all" and click OK
- 3 >> In this window you can see:
  - Printing time estimation
  - Total plunger length used for the print (mm)
  - Amount of material used (mm<sup>3</sup>)
  - Material cost (only if cost are specified in material settings)
- 4 >> Set the preview coloring to "Active Toolhead".
- 5 >> Inspect the preview of the sliced object in the preview window and verify if it appears like expected.
- 6 >> Press the "Export build file to disk" to save the file (.gcode) to a microSD card or save locally to use later.



# 9 Printing with support

Sometimes the printer has a geometry hanging in the air that needs support material to be able to print. This can be the case when you want to print an object with very large overhangs or when printing an object that has a difficult printing geometry.

Follow the steps below to enable support in your object.

- 1 >> Select the support placement icon at the right toolbar in simplify (image 25).  
The support window will pop-up where you've been given two different support setup choices:

## Automatic Support generation

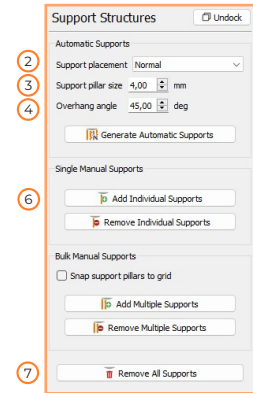
- 2 >> **Support placement:** defines how the support is placed. You are given two options:  
**Normal:** Support will be placed everywhere it is needed (depending on the angle you have defined).  
**From build platform only:** Support will be built only from the build plate, no supports on the model.
- 3 >> **Support pillar size:** define the resolution where support is placed. If you have an edge of 2.0 mm in your model where support is needed, a pillar size of 3.0 mm will not work. (only 2.0mm or smaller)
- 4 >> **Overhang angle:** Defines the angle where support is placed. If an angle is greater than 45 degrees, support will be placed. If smaller than 45 degrees, support will be skipped on that place.

## Manual Support generation

- 5 >> Select the support pillar size (step 2). This is the size the pillars are going to be that you want to place on the model.
- 6 >> Afterwards you can manually place support by pressing the **"add individual supports"** or the **"add multiple supports"** button.
- 7 >> To remove previous placed support, press the **"Remove individual supports"**, **"Remove Multiple supports"** or the **"Remove all Supports"** button.



Image 25



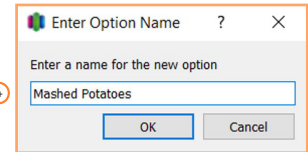
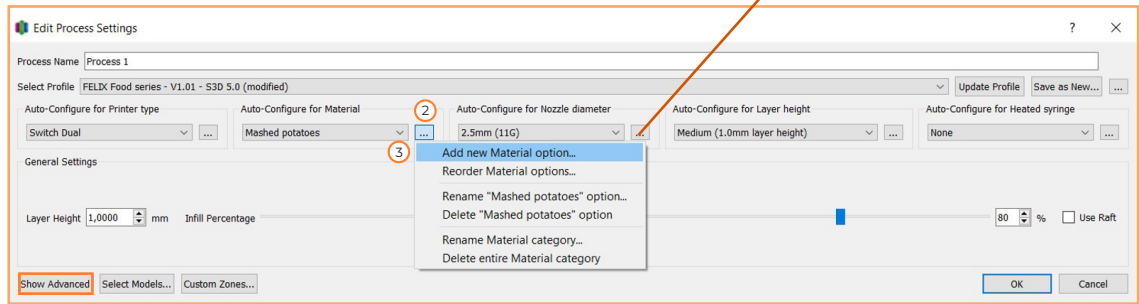
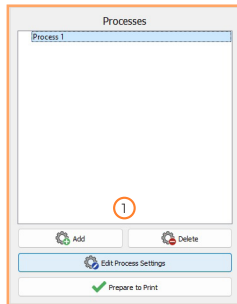


# 10 Adding new Recipes

## 10.1 Add a new material

This chapter will explain how to create and add your own new materials to simplify3D.

- 1 >> Click "Edit process settings" in main window.
- 2 >> Press the 3 dots button next to the material dropdown.
- 3 >> Select "add new material option".
- 4 >> Give the material an appropriate name.
- 5 >> And press the "Show advanced" button to configure the material settings.



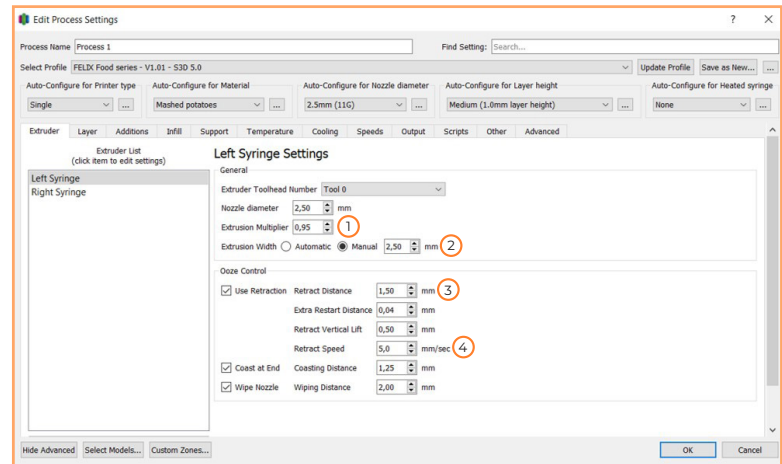
## 10.2 Configure new material settings

In the advanced settings menu we can set up the different parameters according to the newly setup material. In this manual we are going to focus on the three most important parameter tabs (extruder, temperature and other).

We will start with the extruder tab. In this tab you can find everything extruder-related. Only change the left syringe parameters if you are using a Single head. If you have a Switch head or Twin head change the parameters for both left and right syringe.

### Extruder

1. **Extrusion multiplier.** This determines the amount of material that is laid down for each line. The correct value is between 0.9 and 1.1 in general.
2. **Extrusion width.** This determines the width of each line that is laid down. Normally it is the same as your nozzle diameter
3. **Retraction distance.** Some materials will leak out of the nozzle. You can raise or lower the retraction distance a bit to prevent oozing
4. **Retraction speed.** Some materials need a higher retraction speed to make sure less material will leak out of the nozzle.
5. If you have a Switch head make sure you apply all settings you change to both syringes.



In the next tab, the temperature settings are covered. Set temperatures here if you want to heat the build plate or syringes during the print.

Only change the **left syringe** temperature if you are using a **single head**. If you have a **Switch head** or **Twin head** change the temperature for **both left and right** syringe.

## Temperature

1. Select **"Both"** in the heated syringe dropdown to unlock the input field.
2. Change the 0 under temperature to the desired temperature for the first layer.
3. Press heated bed and change the temperature the same way you changed it for the left syringe.
4. If you have a dual head FOODprinter, make sure you apply the desired temperature for the right syringe with the same as you did for the left syringe.

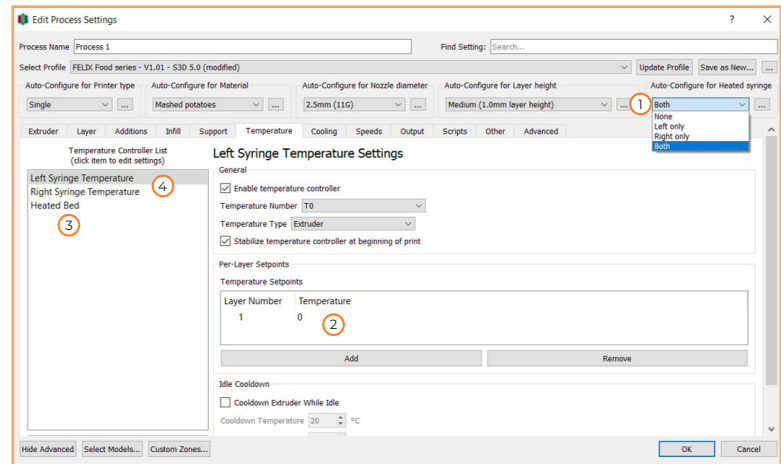
### **!! WARNING !!**

The maximum temperature of the extruders is 80 °C. It will take some time to fully heat up the syringes:

**80 °C** = +/- 60 minutes

**50 °C** = +/- 50 minutes

**30 °C** = +/- 40 minutes.



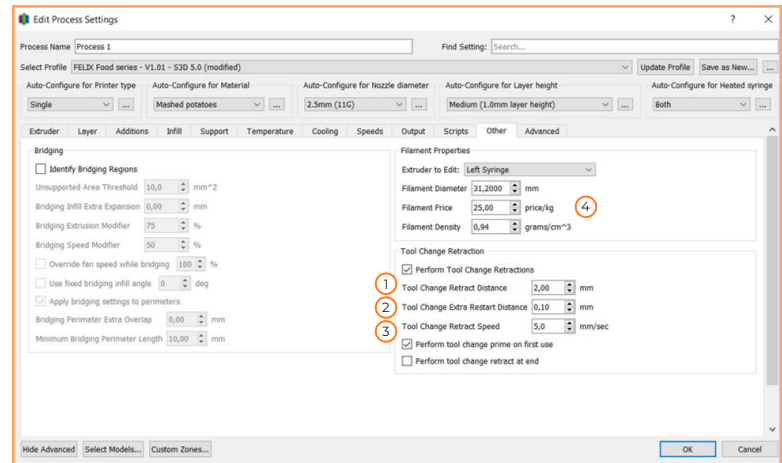
In the other tab we will change the tool change parameters. This is only used in dual head prints with a Switch head. You do not have to change them when using a Single head or Twin head printer. Only change the left syringe temperature if you are using a single head. If you have a Switch head or Twin head change the temperature for both left and right syringe.

## Final Details

1. The tool change retraction distance is used to define the distance that the syringe needs to retract when it's printing with the other syringe.
2. The tool change extra restart distance is an extra amount of material that is extruded when starting with the new syringe.
3. The tool change retraction speed is the speed at which the material is retracted and extruder during a syringe change.

The last step is price and density. This is not necessary to configure the material but is nice to see the correct values in the preview.

4. Fill in the material cost and the material density to get the best results in the preview. Do not change the diameter!
5. The last step to do is saving your settings. Press update profile to save all the setting you made.
6. Press "OK" to finish the configuration.



# 11 3D printing Tips & Tricks

## 5 Steps to a successful print

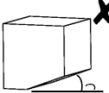

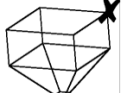

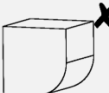



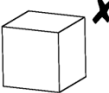

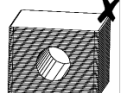
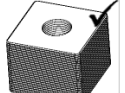


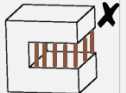
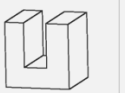
- ✓ **Clean print surfaces.**  
Ensure print-surface is clean. Use recommended detergent suitable for the print-process.
- ✓ **Clean exterior of nozzle.**  
Remove remainders of print material with a paper towel or cloth before starting each print.
- ✓ **Use supplied accessories.**  
The use of supplied accessories ensures optimal machine performance.
- ✓ **Wait for first layer to finish.**  
The success of a print is mostly depending on the first layer. Make sure first layer is finished properly before leaving your printer unattended.
- ✓ **Preheat the extruders at the correct temperature.**  
Before starting a print let the printhead and bed temperature settle on their correct temperatures before printing.

## 5 Steps to a successful print

- ✓ **Make sure build plate is leveled.**  
Especially for larger print objects a properly leveled bed is essential for good first layer adhesion to the build plate.
- ✓ **Use latest slicing profiles and firmware.**  
Make sure to check for updates regularly. Using the latest profiles and settings improves performance.

# 12 3D printing design tips

Besides choosing proper slice settings, life can be made easier if you print 3D objects suitable for 3D printing. The next table shows an overview of best practice when creating your own design.

		Create overhangs greater than 45° with respect to the print surface.			Reorient model, for largest adhesive surface to print surface. Reduce chance to tipping over objects.
		Avoid rounded corners touching the build plate, create a chamfer to have at least an angle of 45°.			Avoid small surface to volume ratio. Use surface area larger than 5x5mm. Ratio (length or width)/height < 1:5, to prevent tipping over during printing.
		Make small fillets on sharp corners to improve print results.			Orient model for maximum strength. Holes are stronger when printed in plane.
		Divide objects in parts to make it printable.			Re-orient part to make sure the object can be printed



### Review our products

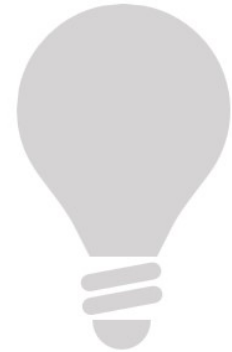
Help us to improve our products  
and services

[www.felixprinters.com/survey](http://www.felixprinters.com/survey)



### Manuals & tutorials

[www.felixfood.nl/contact](http://www.felixfood.nl/contact)  
[www.simplify3D.com](http://www.simplify3D.com)



### FELIX community

[www.felixfood.nl/blog](http://www.felixfood.nl/blog)